eta/DYNAFORM

Die Simulation

Training Manual

Version 5.9.2.1

Engineering Technology Associates, Inc.
1133 E. Maple Road, Suite 200
Troy, MI 48083

Tel: (248) 729-3010
Fax: (248) 729-3020
Email: support@eta.com

Engineering Technology Associates, Inc., ETA, the ETA logo, and eta/DYNAFORM are the registered trademarks of Engineering Technology Associates, Inc. All other trademarks or names are the property of the respective owners.

©1998-2014 Engineering Technology Associates, Inc. All rights reserved.

TABLE OF CONTENTS

TABLE OF CONTENTS **1**

INTRODUCTION **1**

DATABASE MANIPULATION **2**

I. Creating an eta/Dynaform Database and Analysis Setup 2

TOOL PREPARATION **4**

I. Import Die Geometry 4
II. Meshing Die 5
III. Mesh Check and Repair 7

DIE SIMULATION SETUP **11**

I. Die Simulation Interface 11
II. Define Tools 12
III. Define and Mesh Blank 16
IV. Defining the Blank Material 20
V. Define Tool Control 22
VI. Define Drawbead 24
VII. Submit Job 27

POST PROCESSING (with eta/POST) **30**

I. Reading the Results File into the Post Processor 30
II. Animating FLD 32
III. Plotting Single Frames 33
IV. Check the forming Result of FLD 34
V. Check the forming Result of THICKNESS 34

INTRODUCTION

Welcome to the **eta/DYNAFORM 5.9.2.1** Die Simulation Training Manual. The **eta/DYNAFORM** is the unified version of the DYNAFORM-PC and UNIX platforms.

This manual details a step-by-step quick Simulation process through the Die Simulation interface process. Users should take the time to learn these setup processes as it is different from the traditional setup process, such as AutoSetup.

In Die Simulation, use INCSolver to do the sheet metal quick simulation analysis. INCSolver is a nonlinear transient dynamic finite element program using explicit scheme to solve equations of motion, commonly referenced as “incremental code” or “explicit code”. INCSolver uses the SMP (Shared Memory Processing) computing scheme to take full advantage of multiple-CPU, multiple-core and multiple-thread configurations of latest computing platforms in Windows environment.

Note: This manual is intended for the application of all eta/DYNAFORM platforms. Platform interfaces may vary slightly due to different operating system requirements. This may cause some minor visual discrepancies in the interface screen shots and your version of eta/DYNAFORM that should be ignored.

DATABASE MANIPULATION

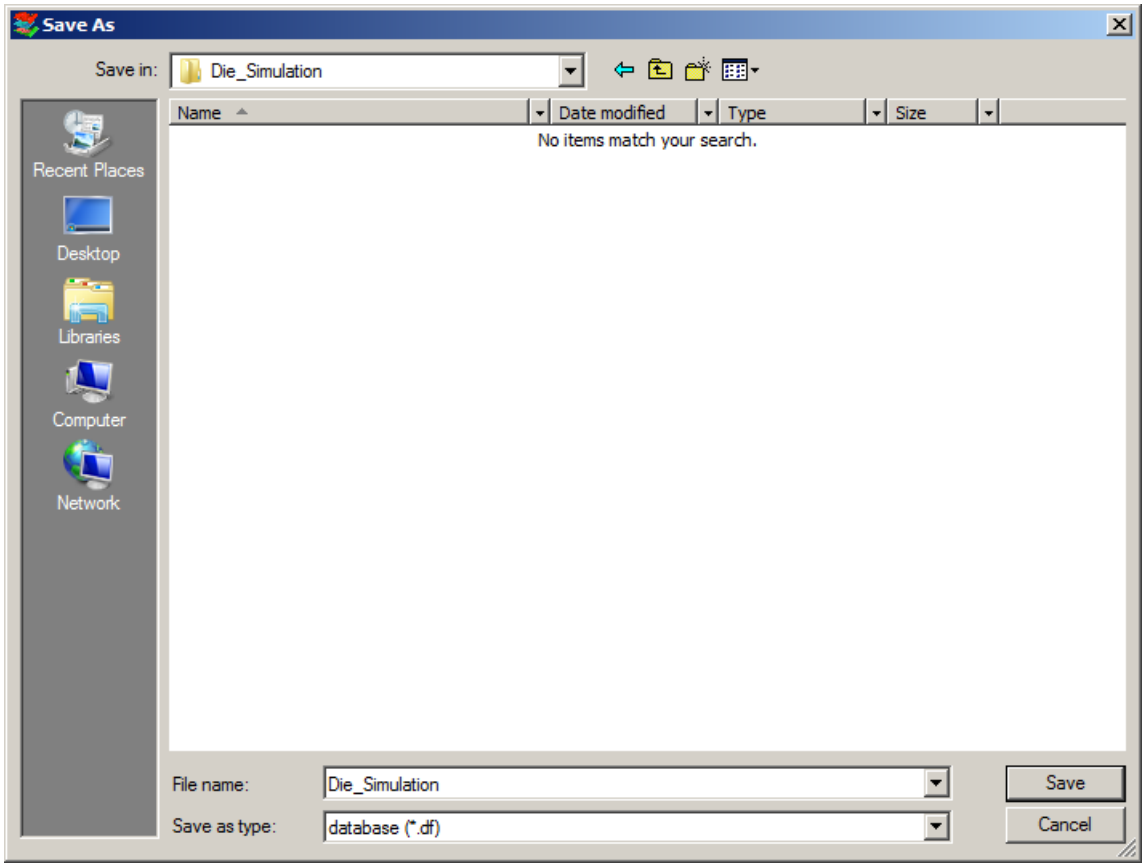
I. Creating an eta/Dynaform Database and Analysis Setup

Start eta/DYNAFORM 5.9.2.1

For workstation/Linux users, enter the command "**df5921**" (default) from a UNIX shell to start eta/DYNAFORM5.9.2.1.

For PC users, double click the **eta/DYNAFORM 5.9.2.1 (DF5921)** icon from the desktop or choose eta/DYNAFORM from the program group.

After starting **eta/DYNAFORM**, a default database file Untitled.df is created. Users can save the database to the designated working directory. Go to **File➡Save as**, type "Die_Simulation", and click **Save** to dismiss the dialogue window.



Database Unit

Click **UserSetup➡Analysis Configuration** menu. The default unit system for a new eta/DYNAFORM database is **mm, Newton, Second and Ton**. The default setting for draw type is **double action (toggle draw)**. The user is able to change these default settings from the **UserSetup➡Analysis Configuration** menu.

Note: Draw Type should accord with press type in practice. The parameters define the working direction of default punch and die. If you are not sure or operating new technique, you should select User Defined.

You can also refer to the eta/DYNAFORM User's Manual for information about all of the Draw Type functions.

File Types

Eta/DYNAFORM has the ability to read the following types of input files:

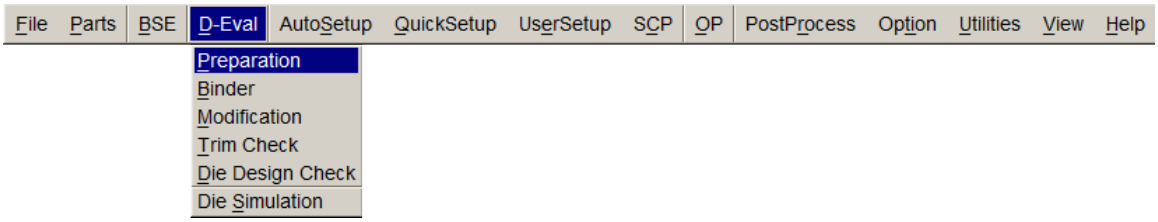
1. IGES （*.igs, *.iges）	10. NX （*.prt）
2. VDA （*.vda, *.vdas）	11. PROE （*.prt, *.asm）
3. LINE DATA （*.lin）	12. INVENTOR（*.ipt）
4. DXF （*.dxf）	13. Parasolid（*.x_t）
5. STL （*.stl）	14. SolidWorks（*.sldprt; *.sldasm）
6. ACIS （*.sat）	15. LSDYNA （*.dyn, *.mod, *.k）
7. CATIA4 （*.model）	16. NASTRAN （*.dat; *.nas）
8. CATIA5 （*.CATPart; *.CATProduct）	17. ABAQUS （*.inp）
9. STEP （*.stp; *.step）	18. DYNAIN （*dynain*）

TOOL PREPARATION

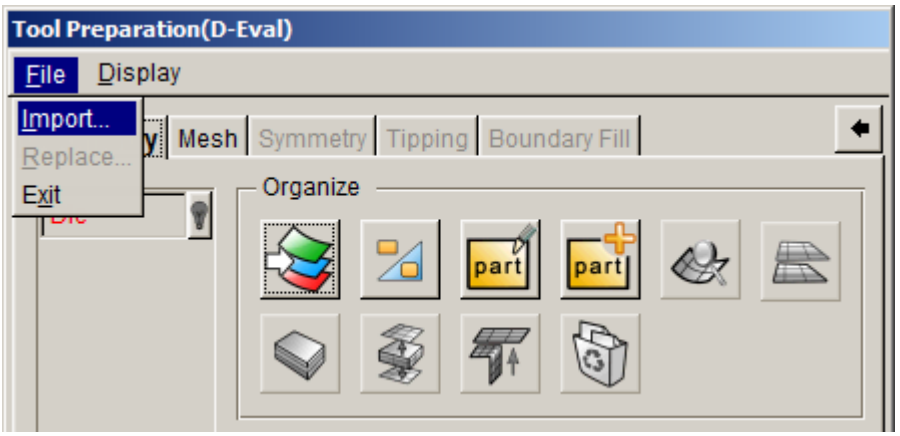
Tool Preparation is a very important step contributing to a successful simulation. The tool will be imported and meshed in Tool Preparation.

I. Import Die Geometry

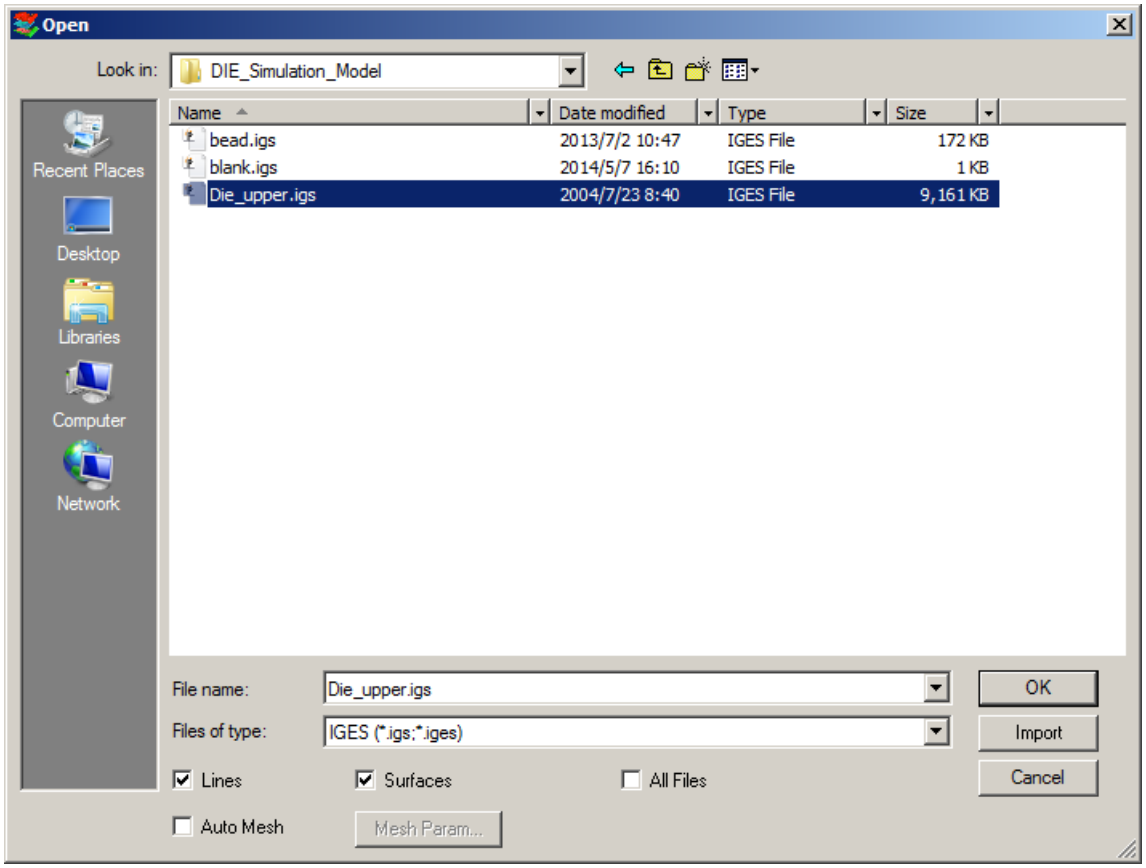
1. From the menu bar, select **D-Eval➟Preparation**.



2. From the menu of Preparation, select **File➟Import.**



3. Go to the training files located. Locate the file: Die_upper.igs, and import it. Click **OK** to dismiss the **Import File** dialogue window.

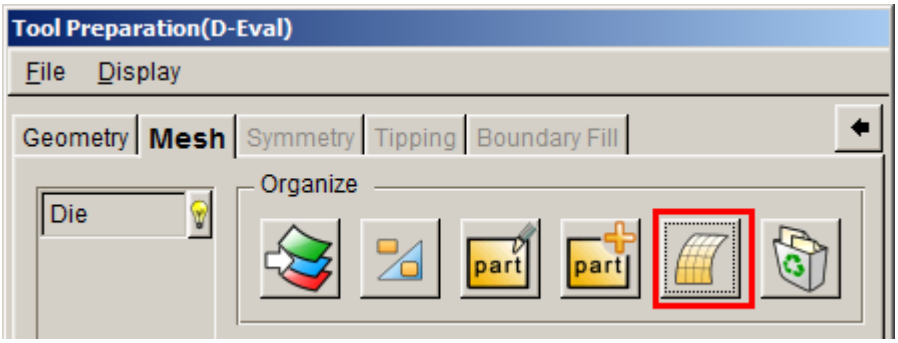


4. After reading in the data file, verify the display looks the same as the illustration shown below.

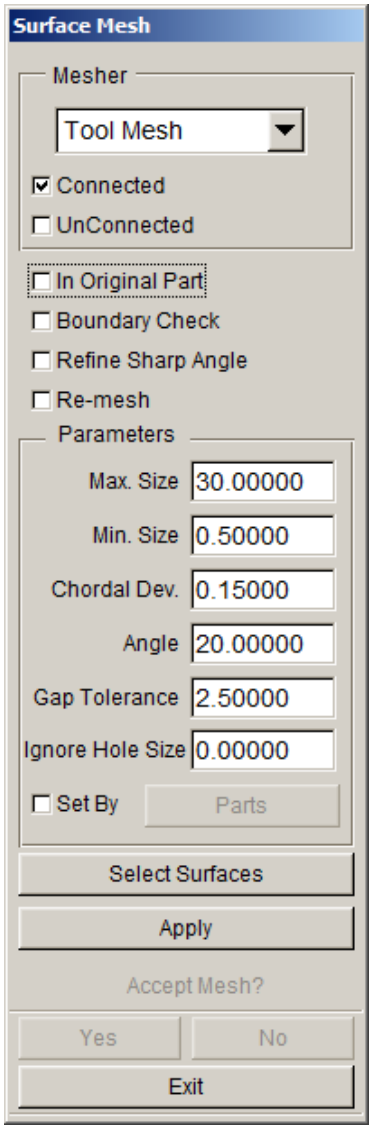
II. Meshing Die

The meshing of the die part tool done in eta/DYNAFORM is carried out using the **Surface Mesh** function. This function will automatically create a mesh based on the provided surface data. This is a very quick and easy way of meshing the tools.

1. Select **Surface Mesh** from the **Mesh** tab as shown below.

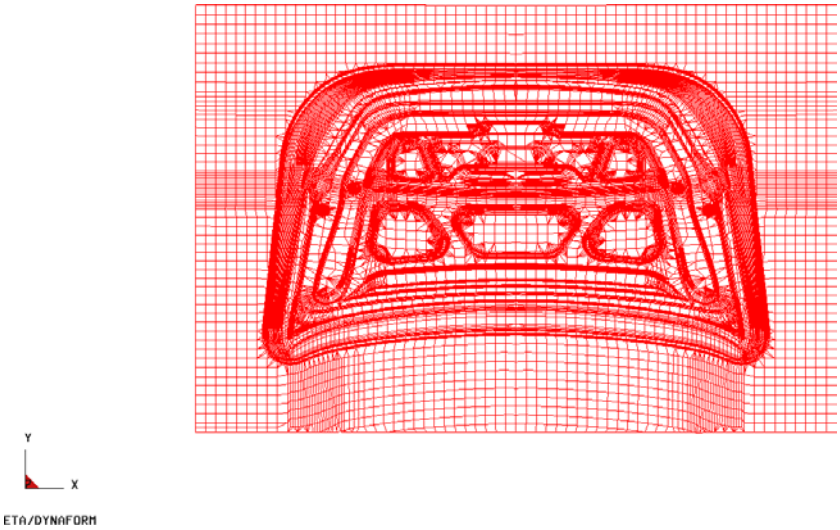


2. In the displayed **Surface Mesh** dialogue window, default values will be used for all fields.



3. All surfaces of **Die** will auto be selected. Click the **Apply** button on the **Surface Mesh** dialogue.

4. The mesh will be created and will be displayed in white. To accept the mesh, click the **Yes** button when prompted, **"Accept Mesh?"** in the **Surface Mesh** dialogue. Check your mesh with the mesh displayed below.



5. Press **Exit** on the **Surface Mesh** dialogue to exit the function.

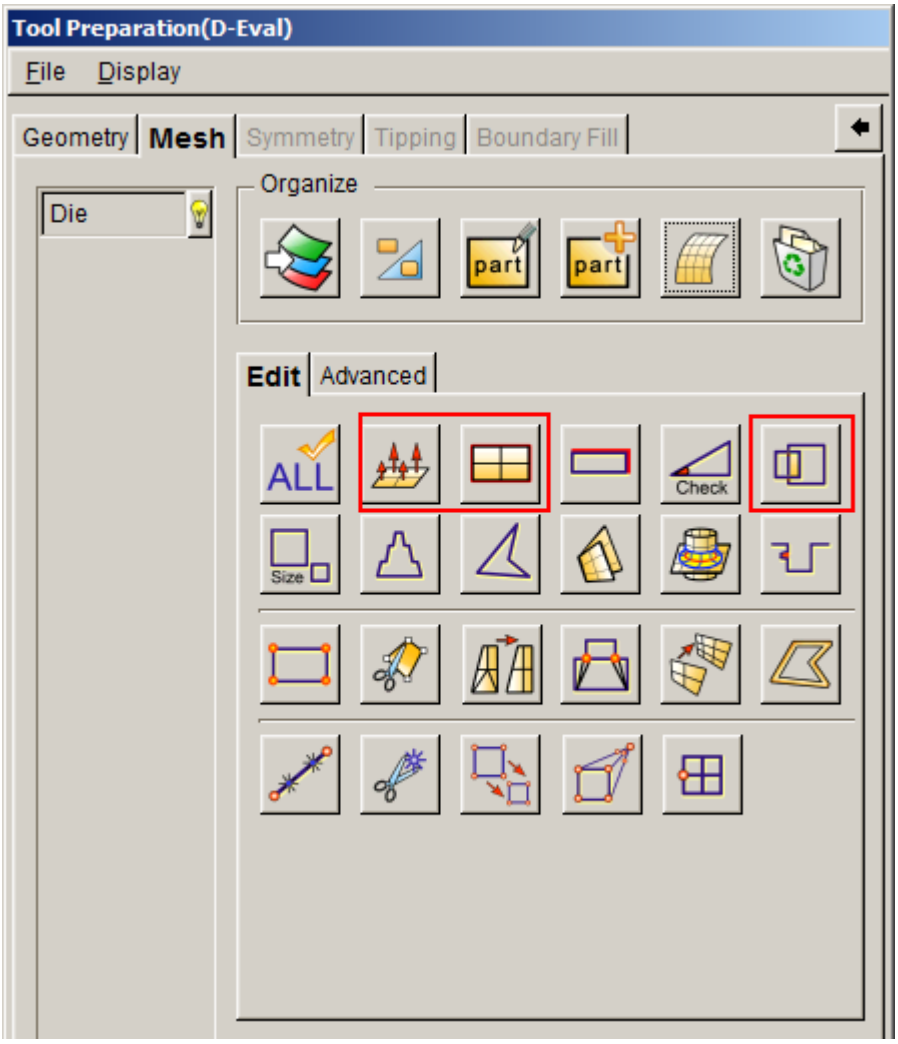
   Now that we have the **Die** meshed, you can turn off the surfaces and lines by turning off **Surfaces** and **Lines** in the **Display Option** dialogue. This makes it easier to view the mesh.

6. Save the database.

III. Mesh Check and Repair

As the mesh has been created, its quality has to be checked to verify that there aren't any defects that could cause potential problems in the simulation.

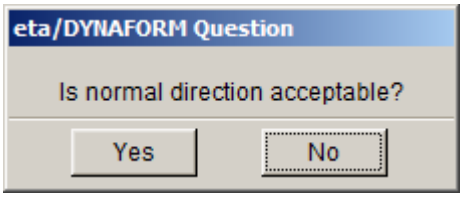
All the utilities used for checking the mesh are located under the **Tool Preparation➡Mesh➡Edit**.

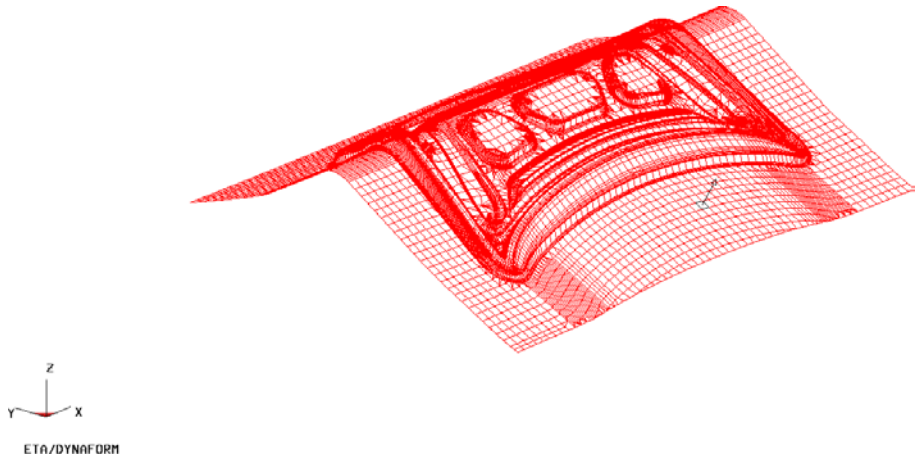


As shown above, the Model Check/Repair dialogue consists of several functions that enable the users to check the quality of mesh. Only two of the functions are described in this training manual. Please refer to **eta/DYNAFORM online help** for information regarding to the remaining functions.

Auto Plate Normal

1. Click **Auto Plate Normal** from the **Tool Preparation/Mesh** dialogue. A new dialogue will be displayed.
2. The displayed dialogue prompts you to pick an element to check all the active parts or an individual part for element normal consistency. Select an element on the **Die part**.
3. An arrow will be displayed showing the normal direction of the selected element. A prompt will ask **"Is normal direction acceptable?"**





Clicking **Yes** will check all elements and reorient the part as needed to the direction that is displayed. Clicking **No** will check all elements and reorient as needed to the opposite of the direction that is displayed. In other words, click **Yes** if you want the normal to point in the direction of the displayed arrow, or **No** if you want it to be the opposite. As long as the normal direction of most elements in a part is consistent, the program will accept that. If half of the total element's normal is pointing upward and other half pointing downward, the result will be flawed.

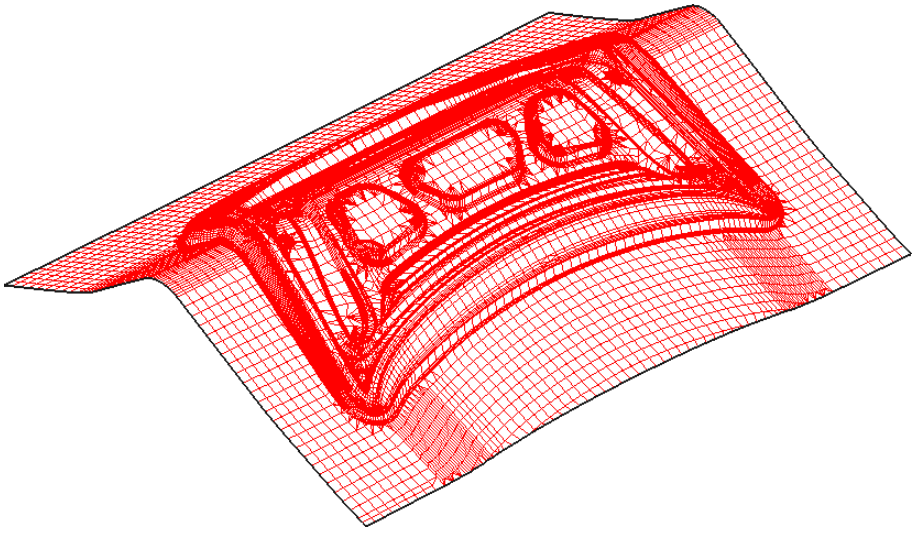
4. Now that the **Die part** elements are consistent.

Display Model Boundary

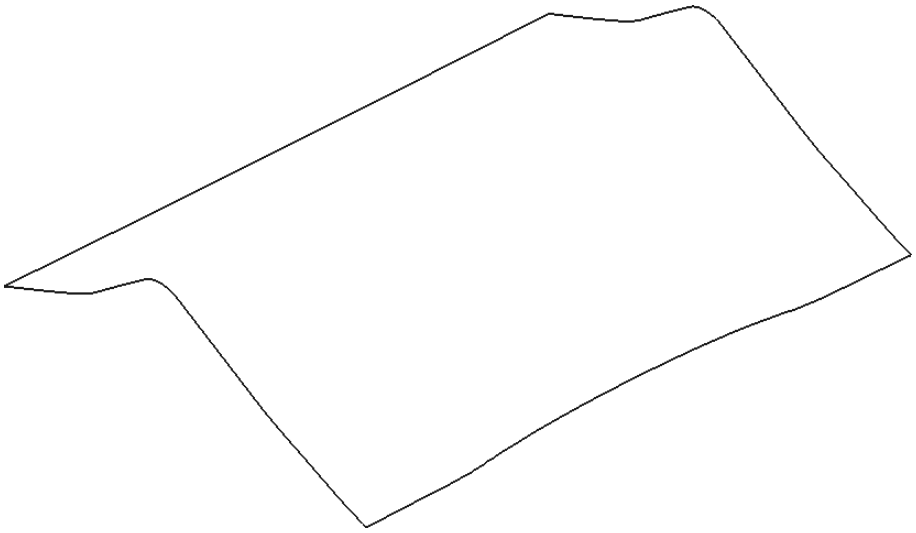
This function will check the mesh for any gaps or holes, and highlight them so you can manually correct the problem.

1. Select **Display Model Boundary** from the **Tool Preparation/Mesh** dialogue window.

Minor gaps in the tool mesh are acceptable. Click the isometric view and make sure that your display looks like the following.



2. Turn off all of the elements and nodes from the **Display Options** dialogue (**Note**: the boundary lines are still displayed). This allows you to inspect any small gaps that might be difficult to see when the mesh is displayed. The results are shown in the following picture.



3. Check for overlapping elements. Delete the duplicate elements if they were found.
4. **Save** your database.

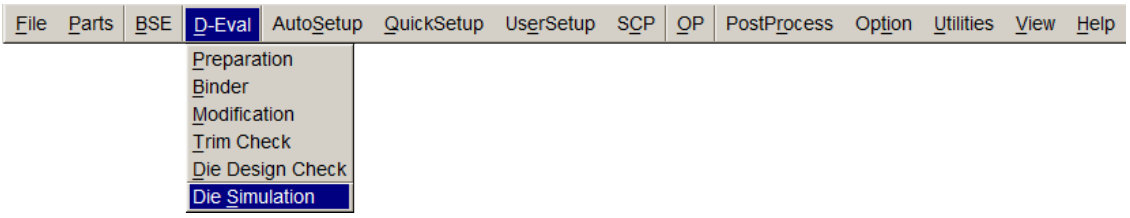
.

DIE SIMULATION SETUP

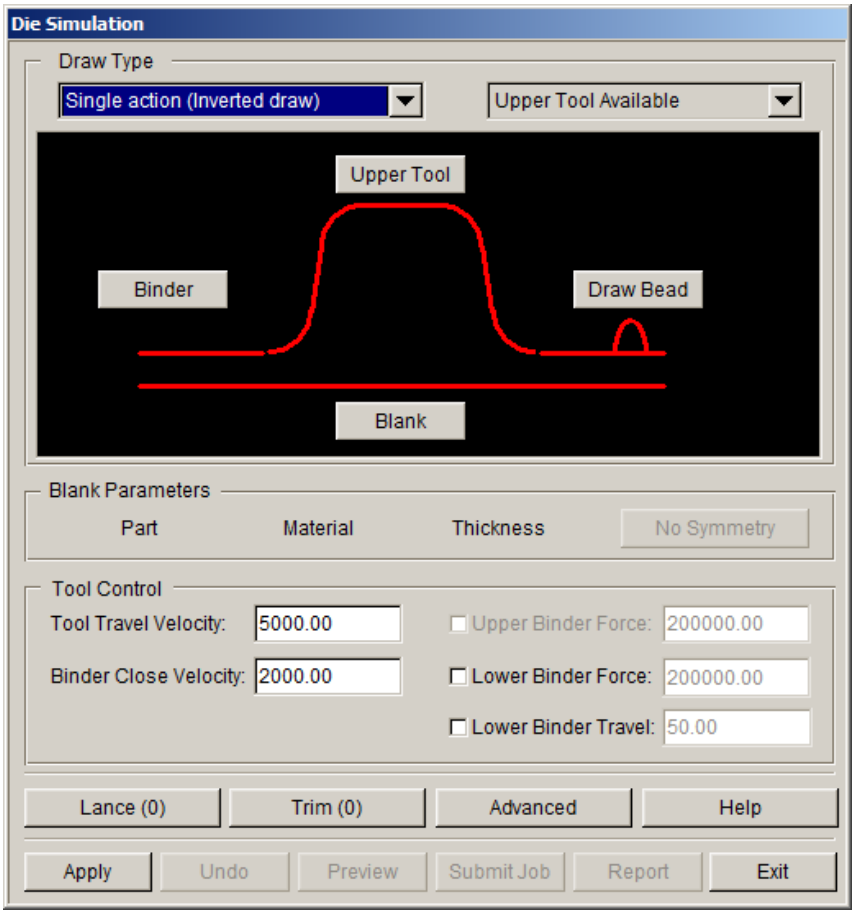
In Die Simulation, use INCSolver to do the sheet metal simulation analysis. Before simulation, it needs to define tools and blank.

I. Die Simulation Interface

1. Select the **D-Eval** menu, **D-Eval➟ Die Simulation.**



2. As shown in the following the Die Simulation menu, the undefined tools are highlighted in red. The user needs to select the **draw type** and **available tool** first. In this application, the draw type is "**Single action**" or **Inverted draw**. The available tool is the **Upper tool**.

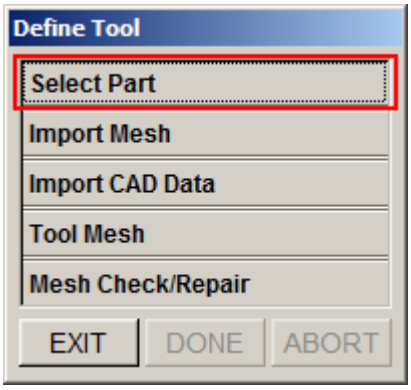


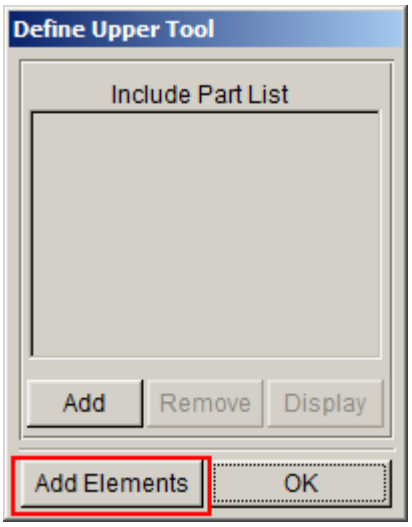
3. Define the tools and blank by clicking the appropriate buttons.

II. Define Tools

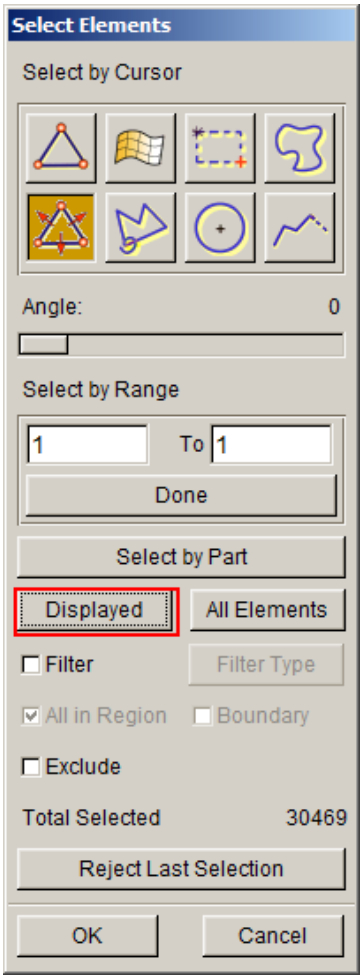
To define the Upper Tool:

1. Click the **Upper Tool** button, and then select the **Select Part** button from the **Define Tool** dialog window**.**

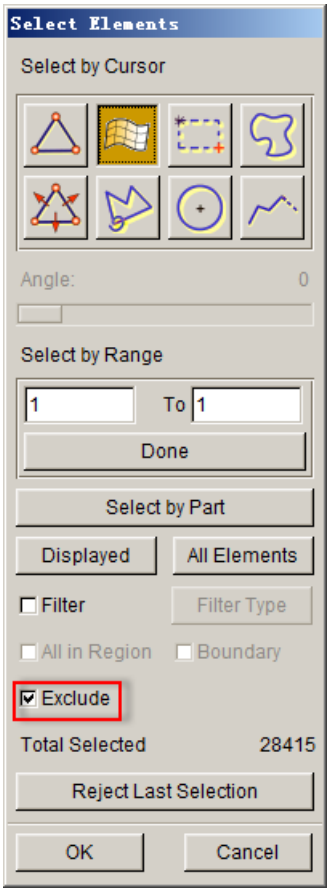


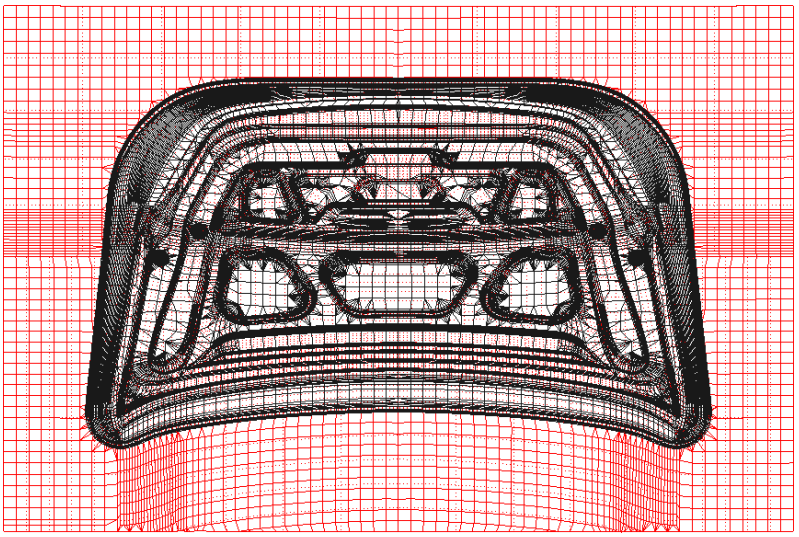


2. Click **Add Elements** from the **Define Upper Tool** window.
3. Click the **Display** button in the **Select Elements** dialog box to select all elements.

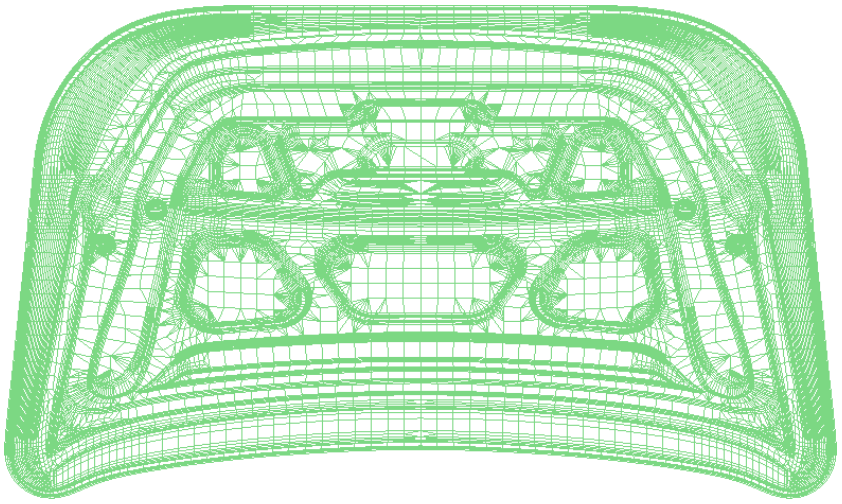


4. Toggle on the **Exclude** option and select the surfaces on binder surface by using **Select by surface**. The elements on binder surface are excluded.



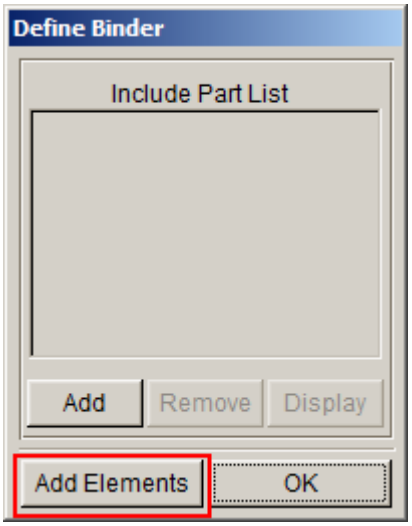


5. The **Upper Tool** is defined. Click **Exit** to exit **Define Tool**.

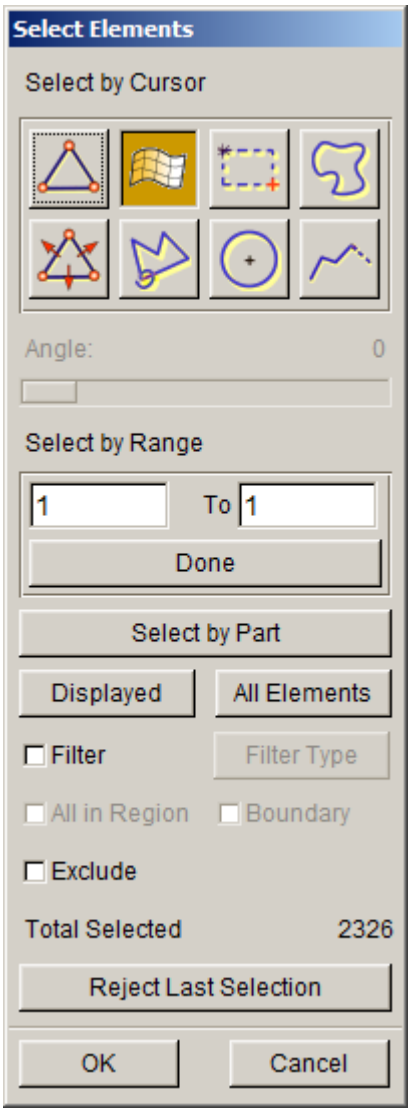


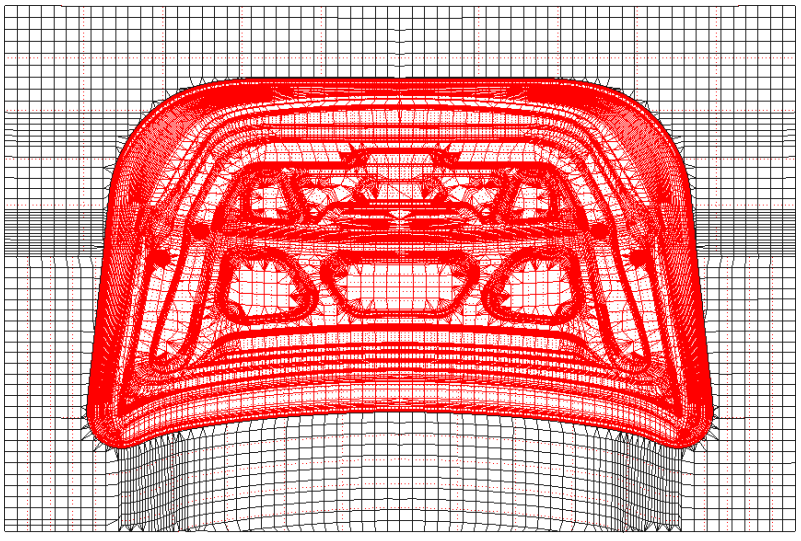
To define the **Binder**:

1. Click the **Binder** button, and then select the **Select Part** button from the **Define Tool** dialog window**.**
2. Click **Add Elements** from the **Define Tool** window.

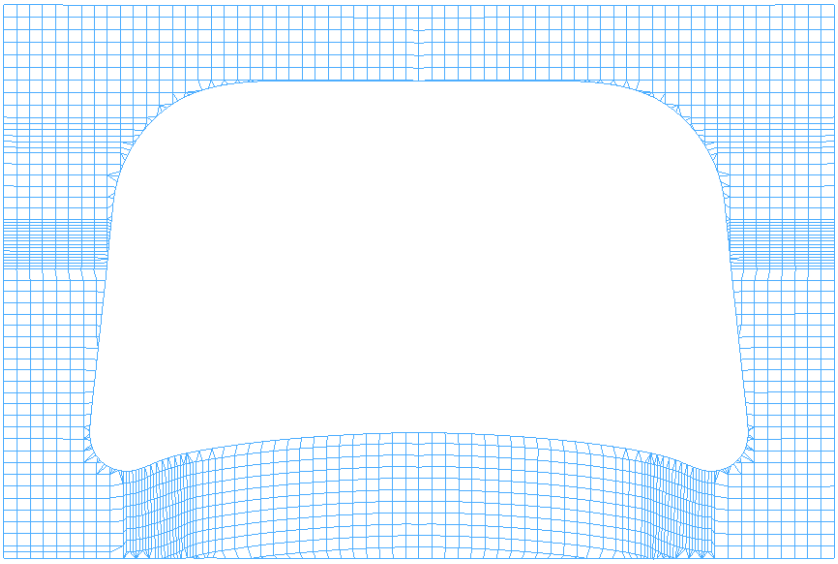


3. Select the surfaces on binder surface by using **Select by surface**. The elements on binder surface are selected.



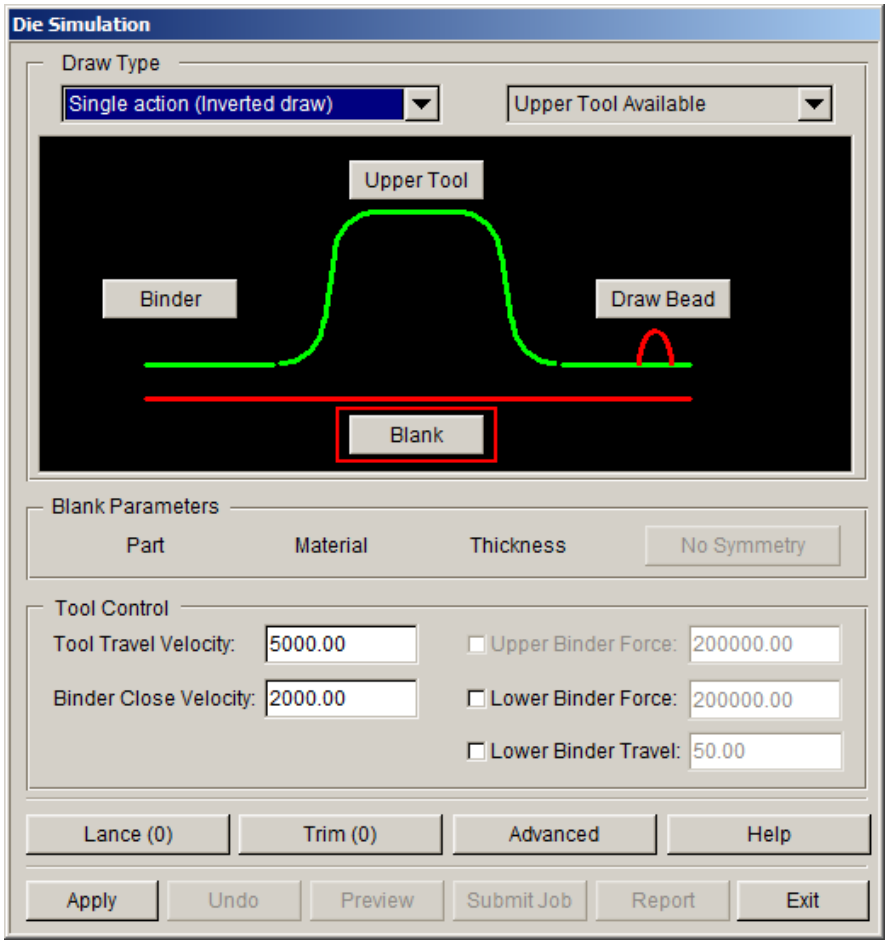


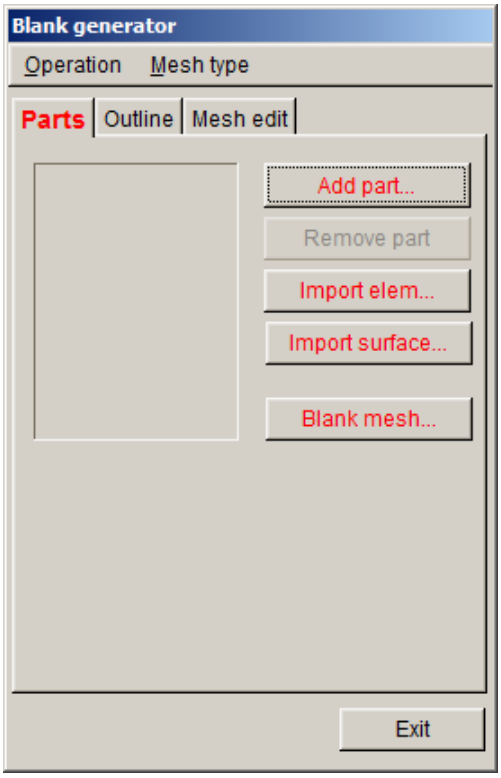
4. The **Binder** is defined. Click **Exit** to exit **Define Tool**.



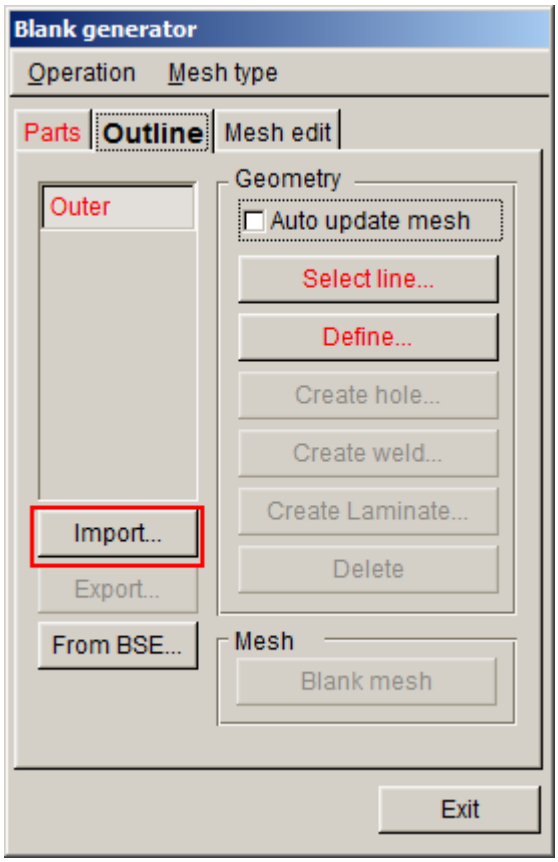
III. Define and Mesh Blank

1. Click the **Blank** button to display **Blank Generator** dialog box.

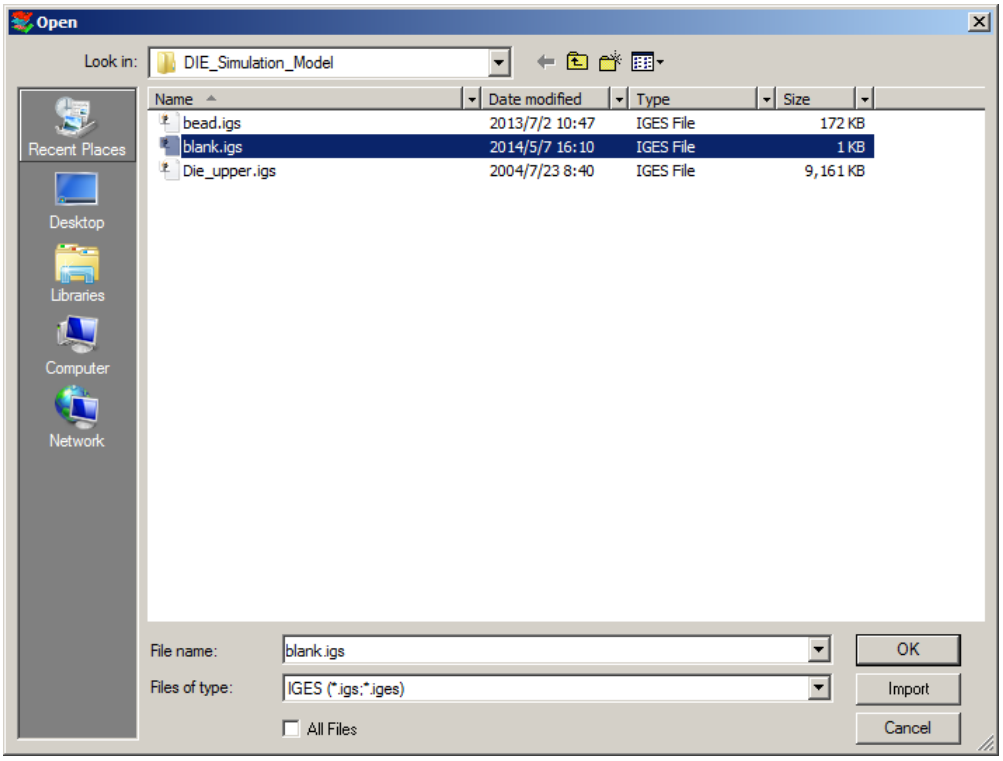




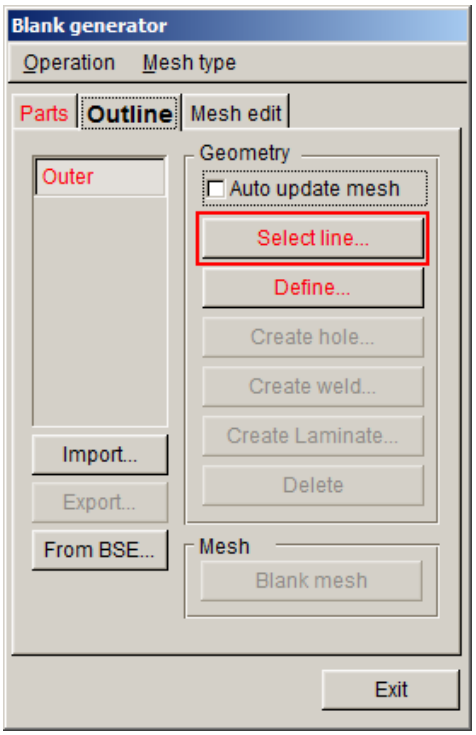
2. In the **Blank Generator/Outline,** click the **Import** button to import the blank outline.

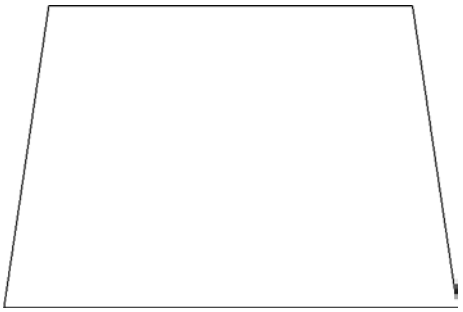


3. Go to the training files located. Locate the file: Blank.igs, and import it. Click OK to dismiss the Import File dialogue window.



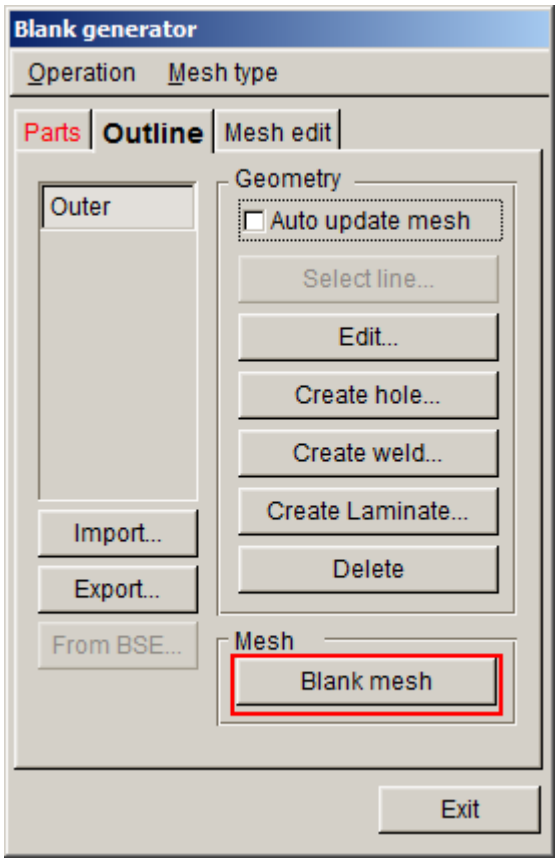
4. Click the **Select Line** button in **Blank Generator/Outline** to select the imported blank outline.



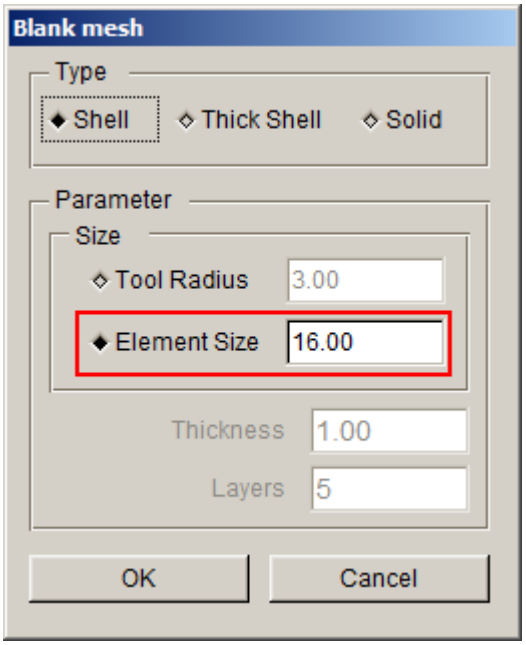


5. After selection, click **OK** to exit Select Line dialog box.

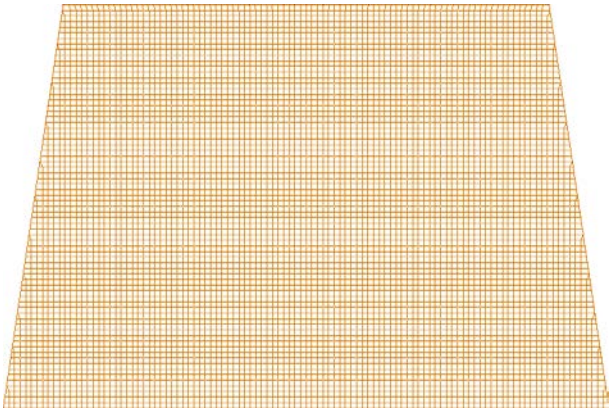
6. Click the **BLANK MESH** button to enter the **Blank Mesh** dialog box.



7. User can change Element Size to 16.



8. After you have entered the variable, press **OK**, and the program will automatically save the created blank mesh as another BLK part. Compare your mesh with the following picture.



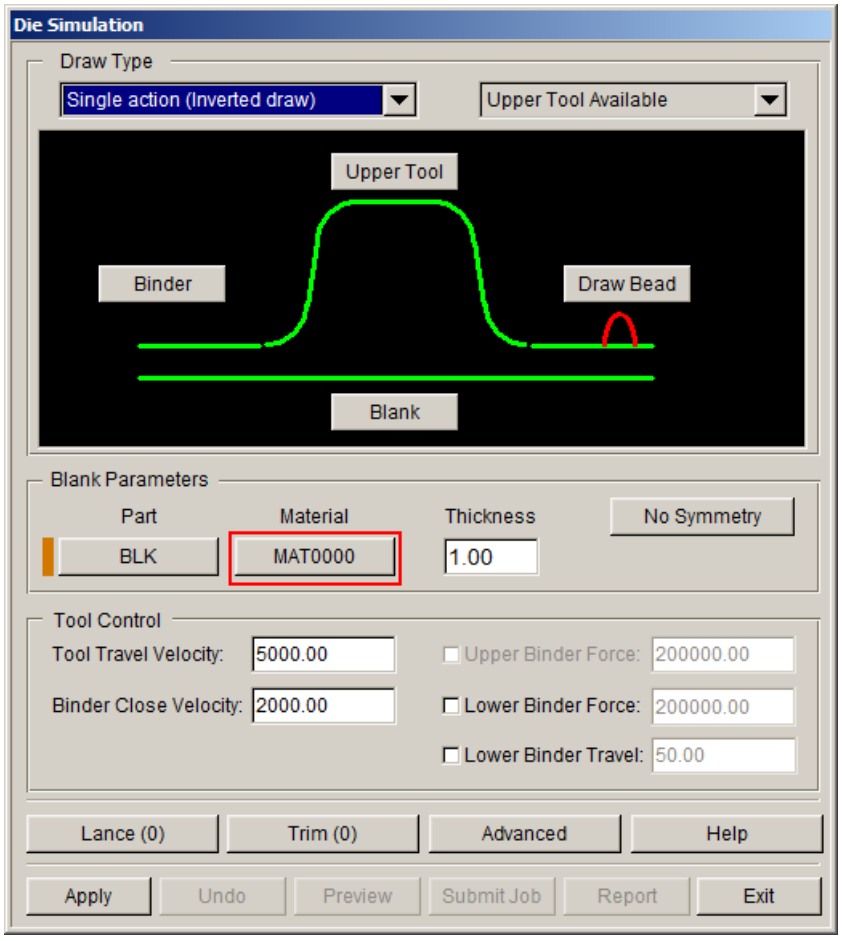
9. **Save** the database.

IV. Defining the Blank Material

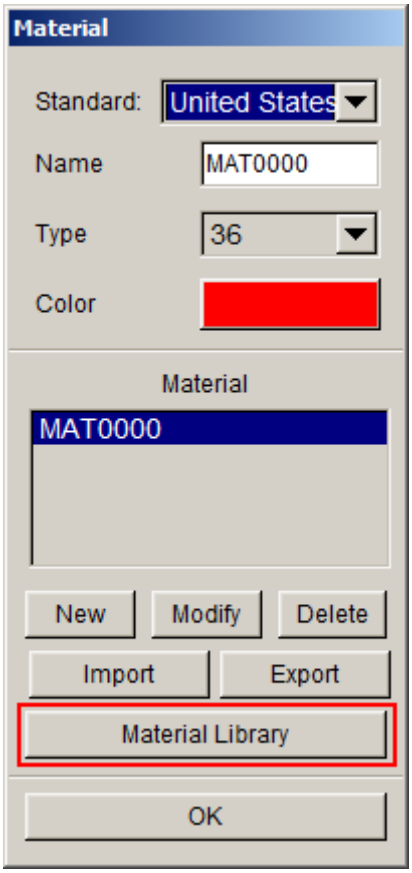
The user needs to define the material and thickness. For the blank thickness, enter the number in the thickness field. In this case, we will set it to 0.8mm.

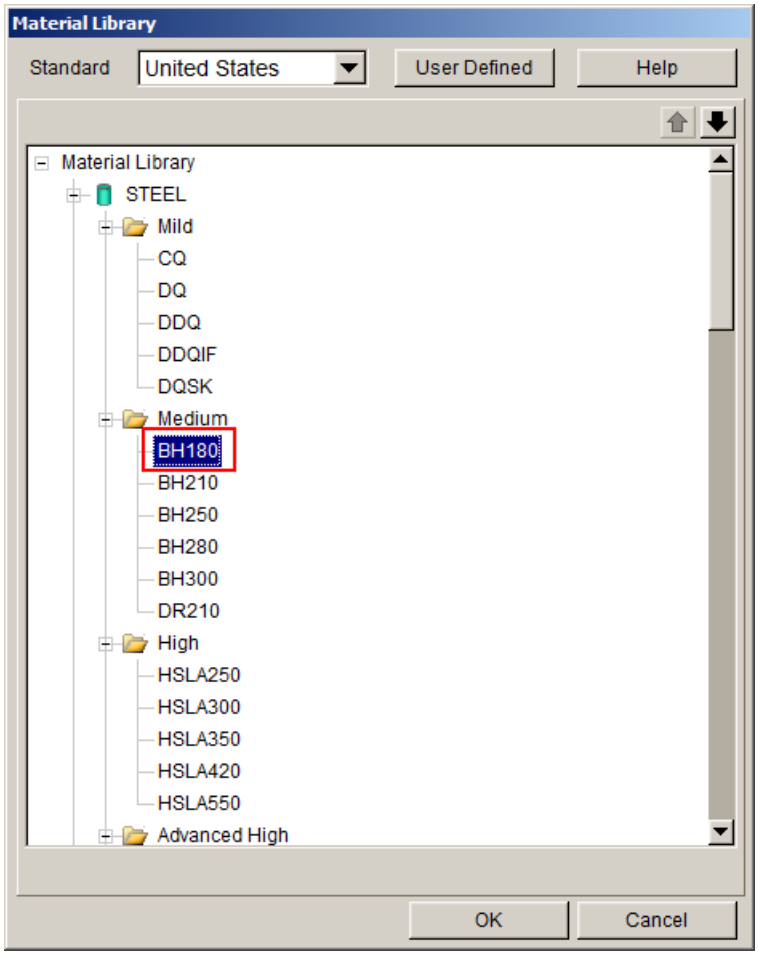
The blank material can be selected from the **Material Library** under the material definition window.

1. Click the **MAT0000** button under Material text.

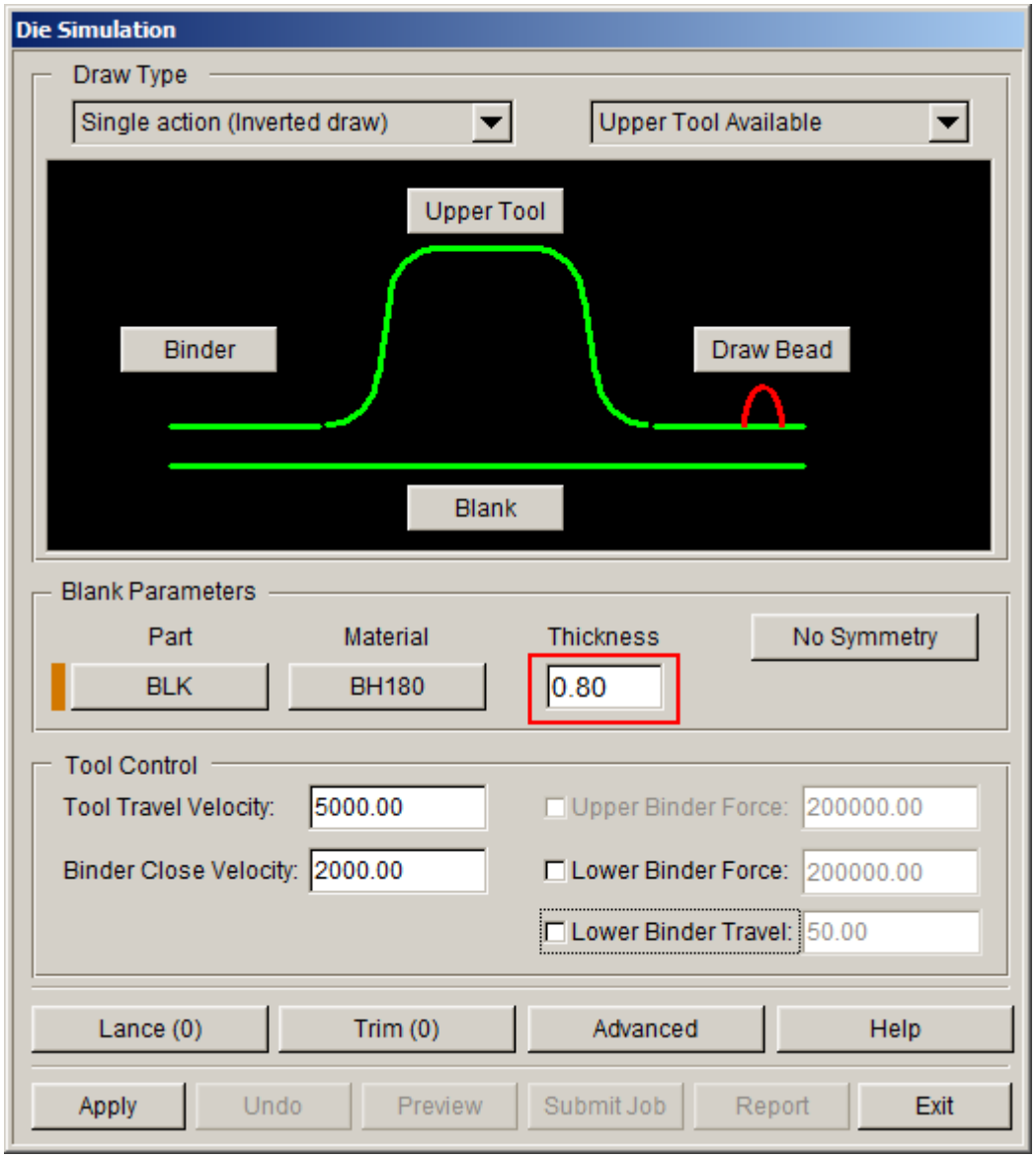


2. Click the **Material Library** button to open the material lib.



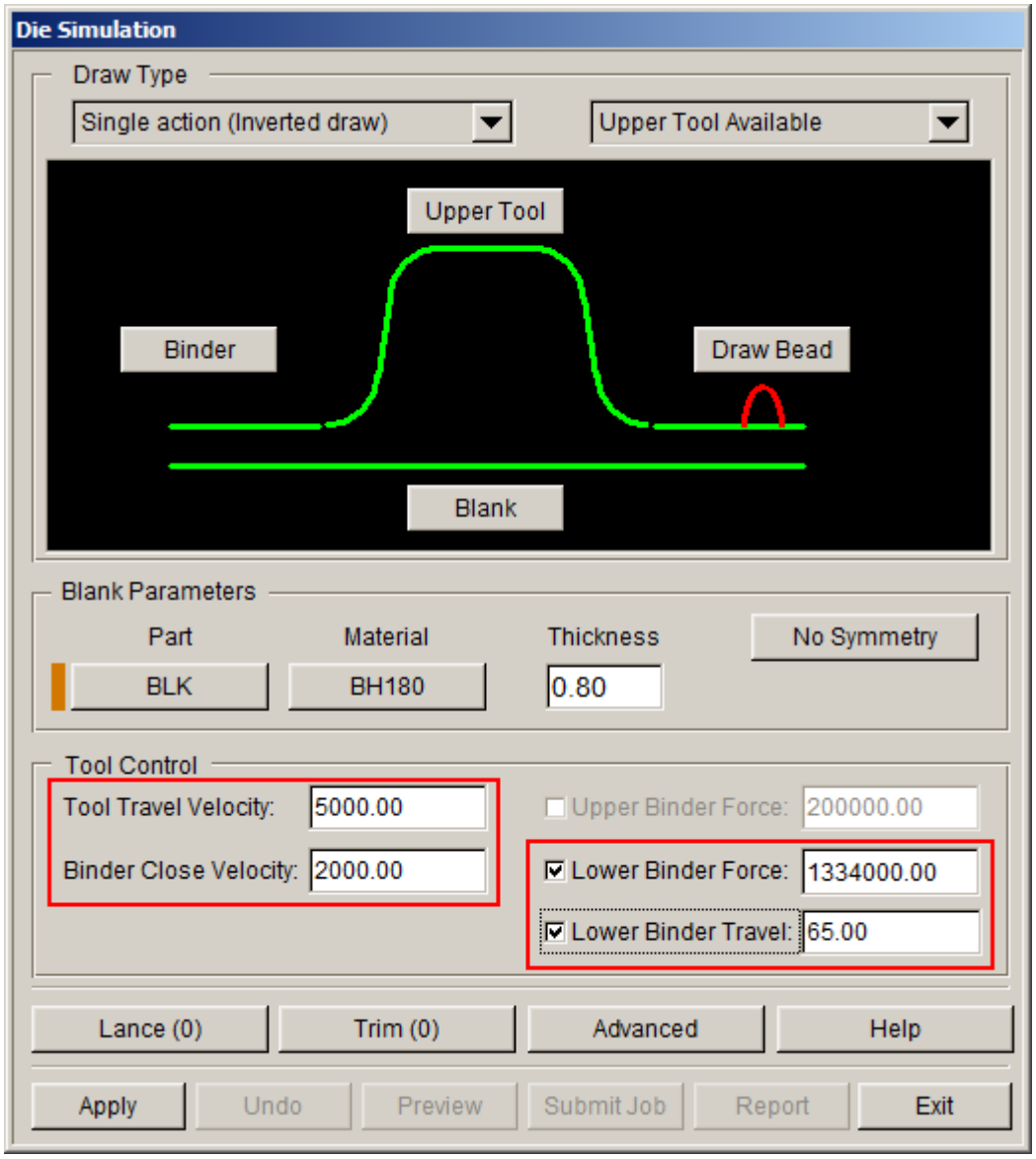


3. Select **BH180** from the Unite States standard. Then exit Material definition dialog box.
4. Input blank thickness: 0.8mm.

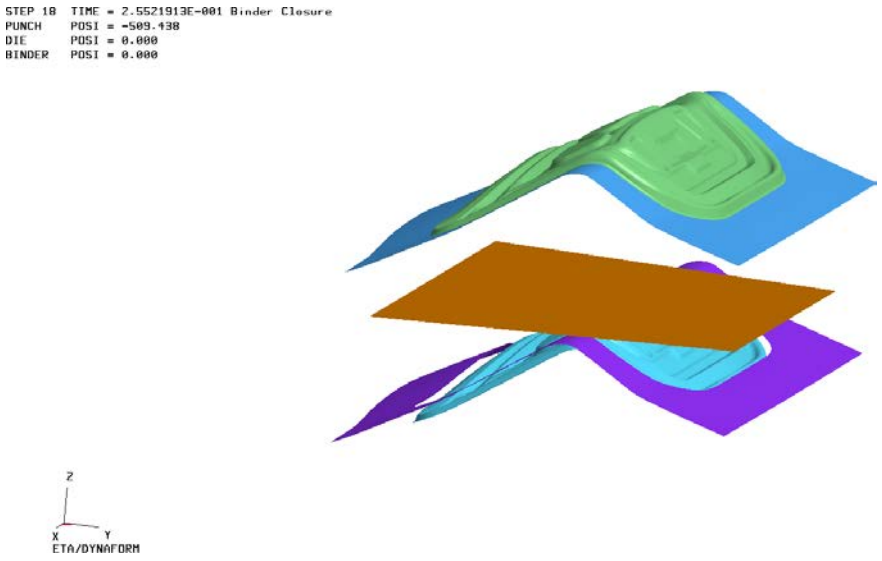


V. Define Tool Control

1. Use the default Tool Travel Velocity and Binder Close Velocity.
2. Toggle on the Lower Binder Force and Lower Binder Travel. Set the value1334000N for Lower Binder Force. Set the value 65mm for Lower Binder Travel, as in the following figure.



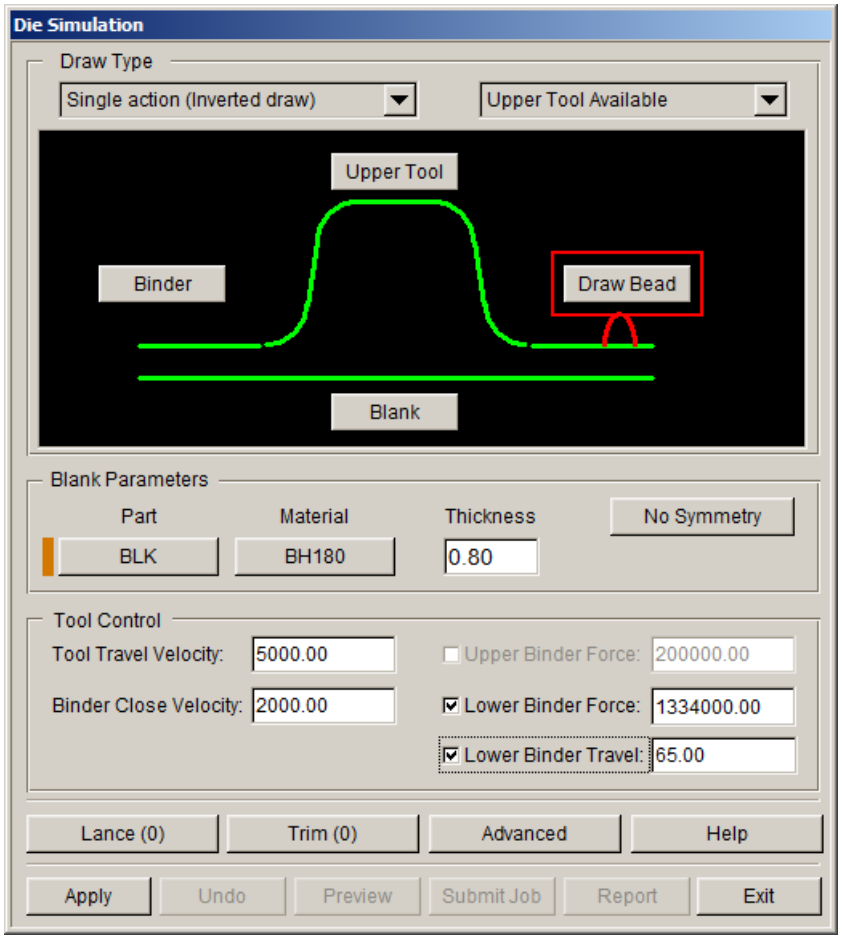
3. Select **Apply**, and the program will automatically create mating tools, position the tools and generate the corresponding travel curves.
4. Select **Preview** to check the tooling motion.
5. Compare your display with the illustration shown below.

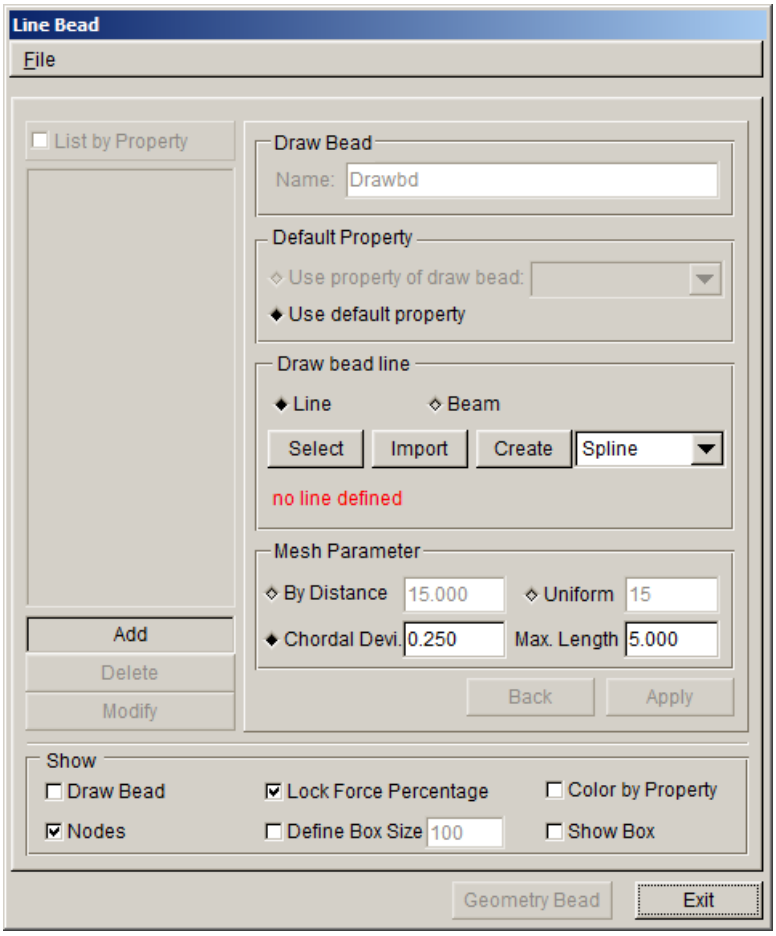


VI. Define Drawbead

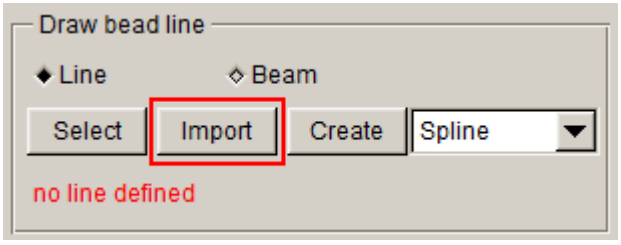
Next step is to define the drawbead for the simulation analysis.

1. Click the **Drawbead** button to open the Drawbead dialog box, as illustrated in the following figure.

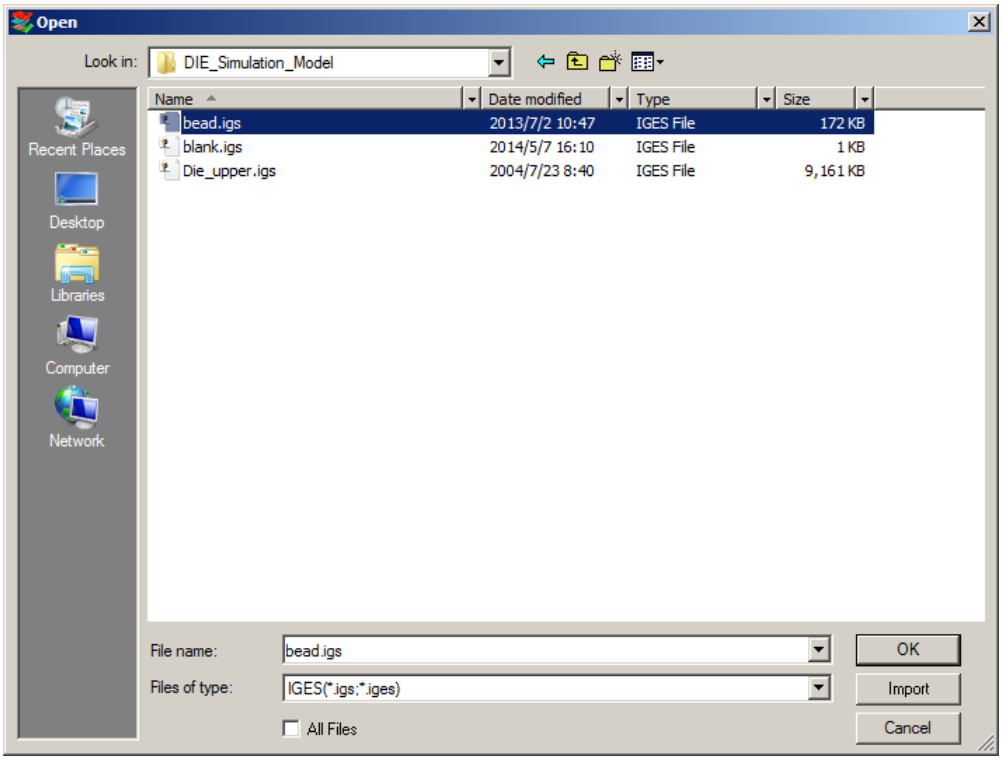




2. Click the **Import** button, as illustrated in the following figure.

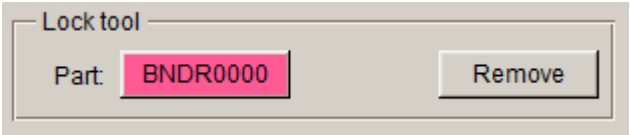


3. Switch the file type to iges and select the drawbead file **bead.igs**.

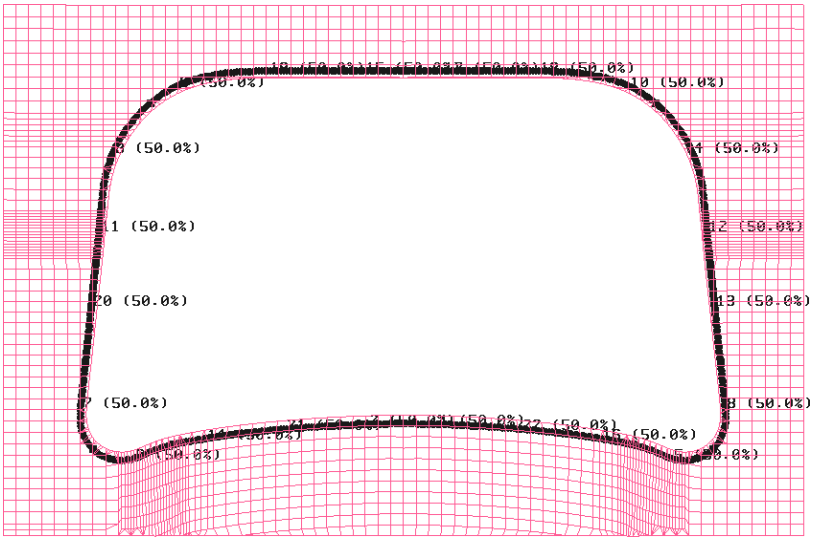


4. Use the default mesh parameters and click **Apply** to generate the line drawbead..

5. Select all the Draw beads in the left side list. Click **Select** in **Lock tool** and select BNDR0000 from the Select Part list. Click OK to exit. Lock the drawbead to part BNDR0000.



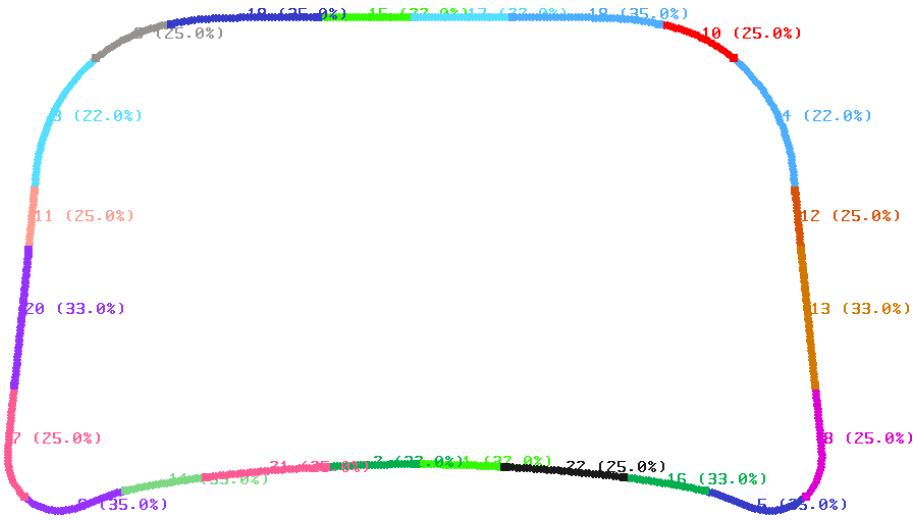
6. Once the sheet is defined, the program will calculate the full lock force and set Force Percentage 50.0% of each bead.



7. Set force percentage for each bead. The force percentages of all drawbeads is in the following table.

Bead ID	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22
Percentage	32	32	22	22	35	35	25	25	25	25	25	25	33	33	22	33	22	35	35	33	25	25

.



8. Click **Exit** to exit the Drawbead dialog box.

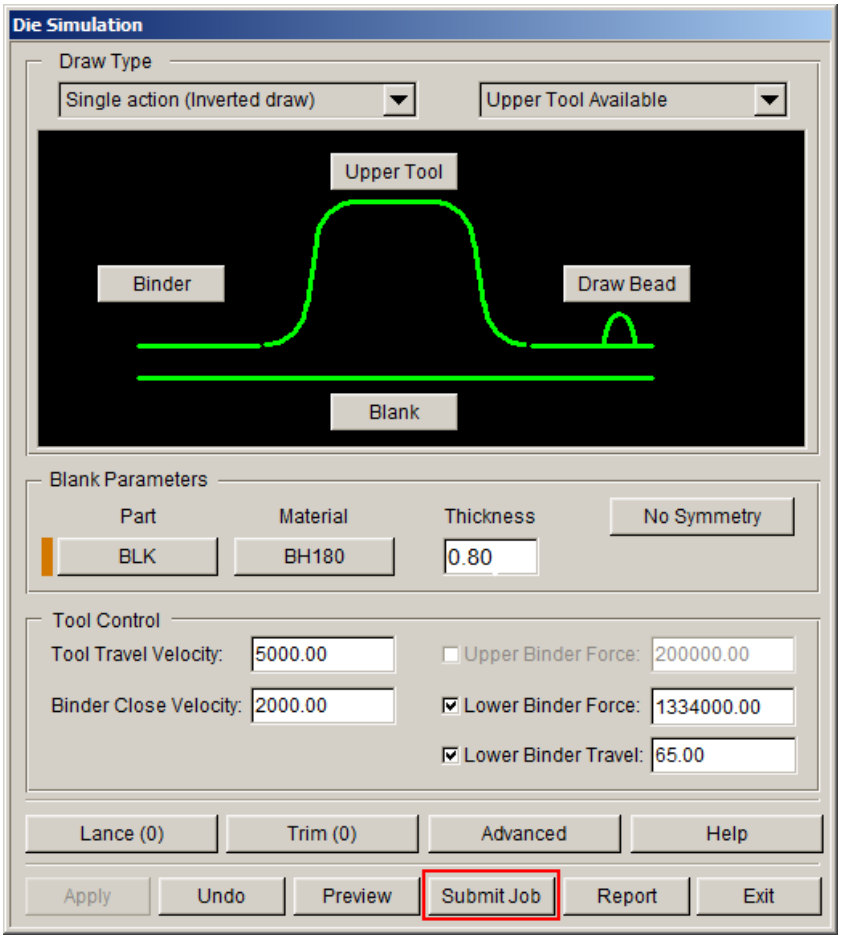
The followings are the description of other functions in the **Die Simulation** interface:

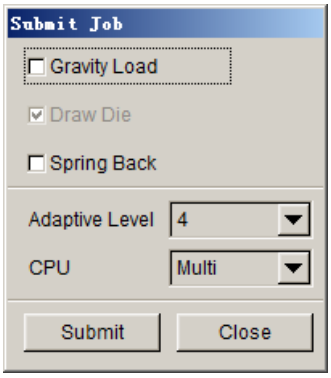
- **Lance and Trim** allows the user to define Lancing and Trimming stages.
- **Advanced** allows the user to change default parameters related to Die Simulation.
- **Help** brings some tips for the user about the Die Simulation.
- **Submit Job** brings the user to the analysis menu.
- **Exit** will allow the user to exit the Die Simulation menu.

VII. Submit Job

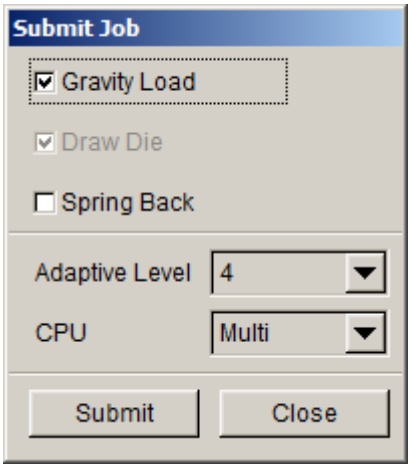
After verifying the tool motion is correct, we can define the final parameters and run the analysis.

1. Click the **Submit Job** button to display the **Submit Job** dialogue window shown below.

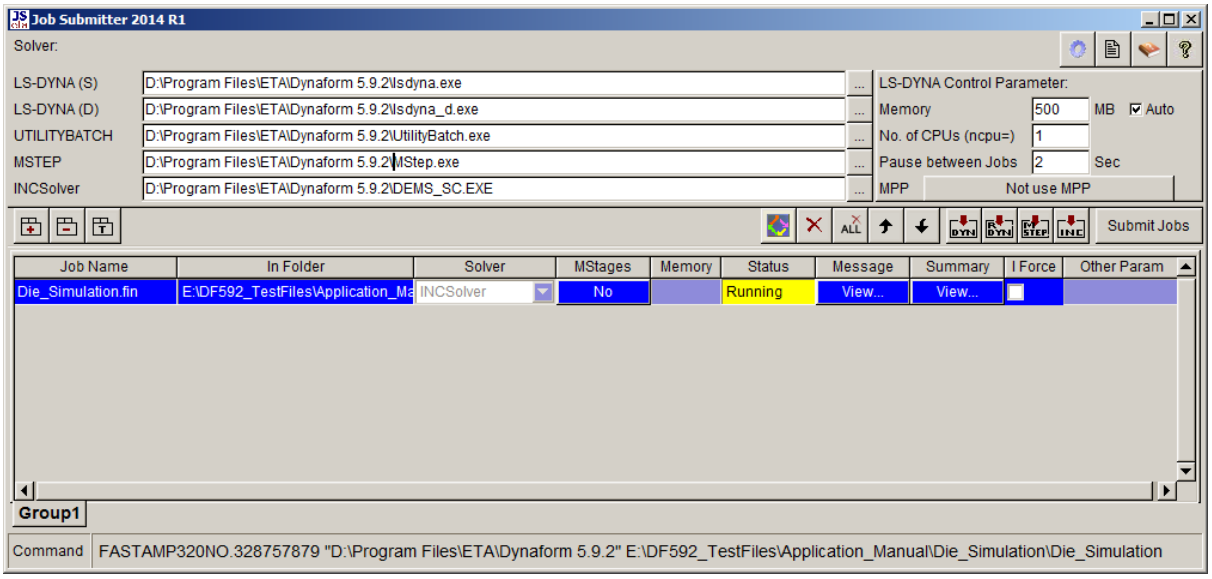




2. Toggle on **Gravity Load** to perform gravity loading analysis prior to Drawing analysis.
3. Set the Adaptive level to **4** and Select Multi from the CPU type button, as shown below.

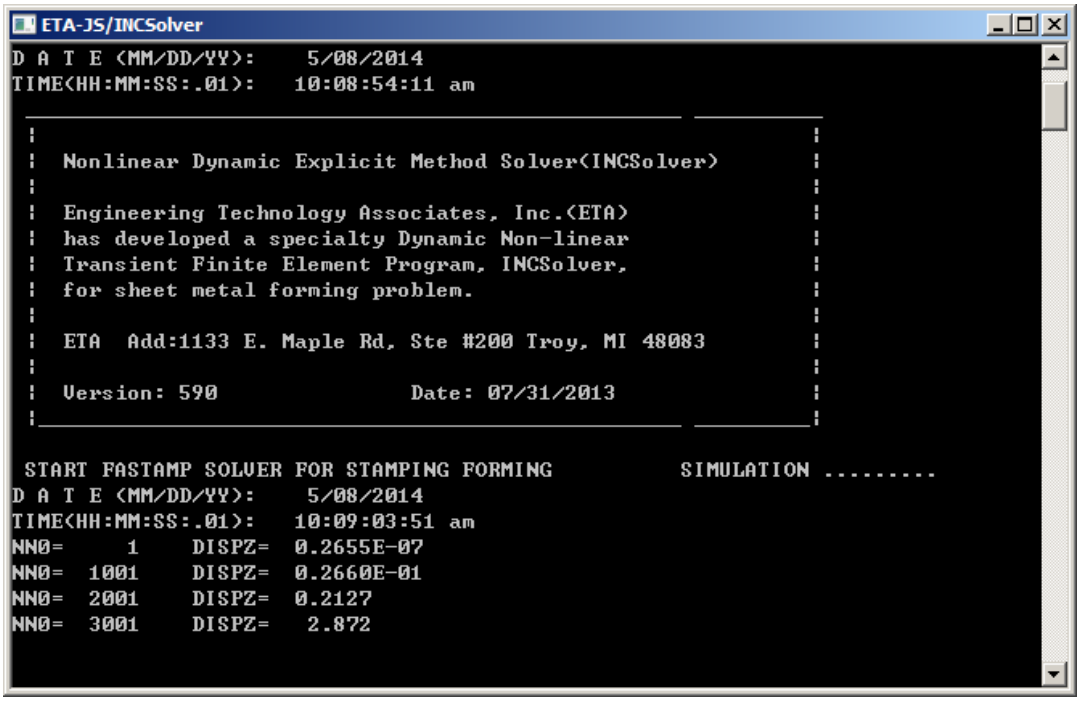


4. Click the **Submit** button to run the job**.**



The INCSolver displays a DOS window, showing the status of the job. You will notice that some model information is given.

The next step is to Proceed to section *POST PROCESSING (with eta/POST)* for post-processing.


ETA-JS/INCSolver
D A T E (MM/DD/YY): 5/08/2014
TIME(HH:MM:SS:.01): 10:08:54:11 am
Nonlinear Dynamic Explicit Method Solver(INCSolver)
Engineering Technology Associates, Inc.(ETA)
has developed a specialty Dynamic Non-linear
Transient Finite Element Program, INCSolver,
for sheet metal forming problem.
ETA Add:1133 E. Maple Rd, Ste #200 Troy, MI 48083
Version: 590 Date: 07/31/2013
START FASTAMP SOLVER FOR STAMPING FORMING SIMULATION .....
D A T E (MM/DD/YY): 5/08/2014
TIME(HH:MM:SS:.01): 10:09:03:51 am
NN0= 1 DISPZ= 0.2655E-07
NN0= 1001 DISPZ= 0.2660E-01
NN0= 2001 DISPZ= 0.2127
NN0= 3001 DISPZ= 2.872

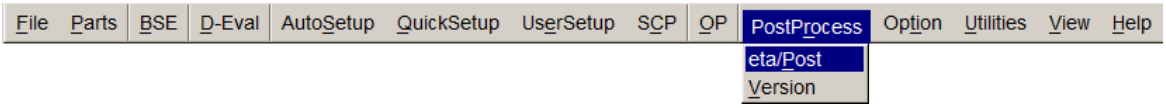
POST PROCESSING (with eta/POST)

The eta/POST reads and processes all the available data in the **d3plot** file (For more information about them, please refer to ***eta-post User's Manual)***. In this case, we will just read the **fas** file which is generated from **INCSolver**.

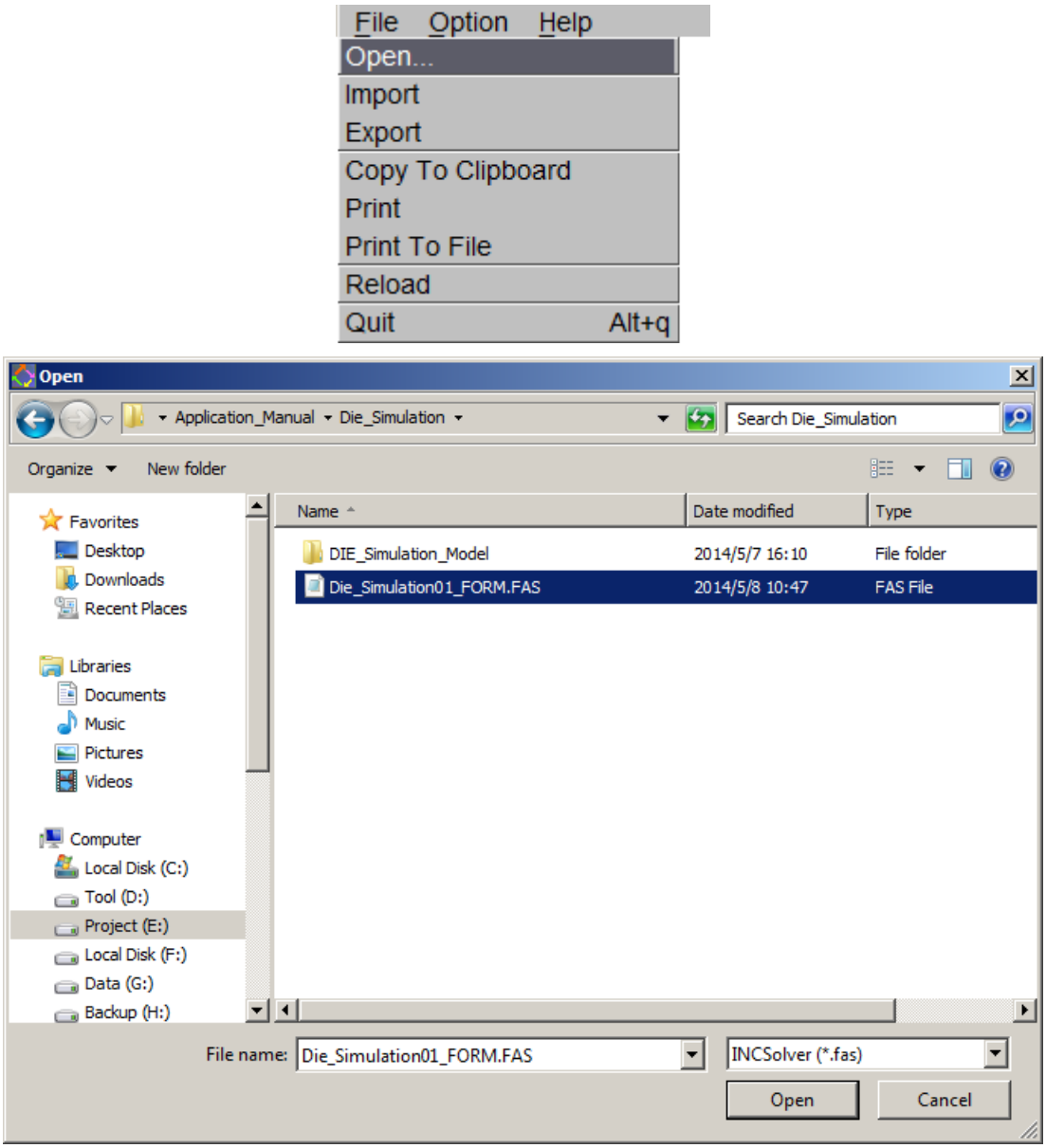
I. Reading the Results File into the Post Processor

To execute **eta/POST**, click **Post-Process** from **eta/DYNAFORM** menu bar. The default path for **eta/POST** is C:\Program Files\Dynaform 5.9.2.1. In this directory, double click the executable file, **EtaPostProcessor.exe**. The **eta/POST** can also be accessed from the programs listing under the start menu under **eta/DYNAFORM 5.9.2.1**.

The eta/DYNAFORM Menu Bar

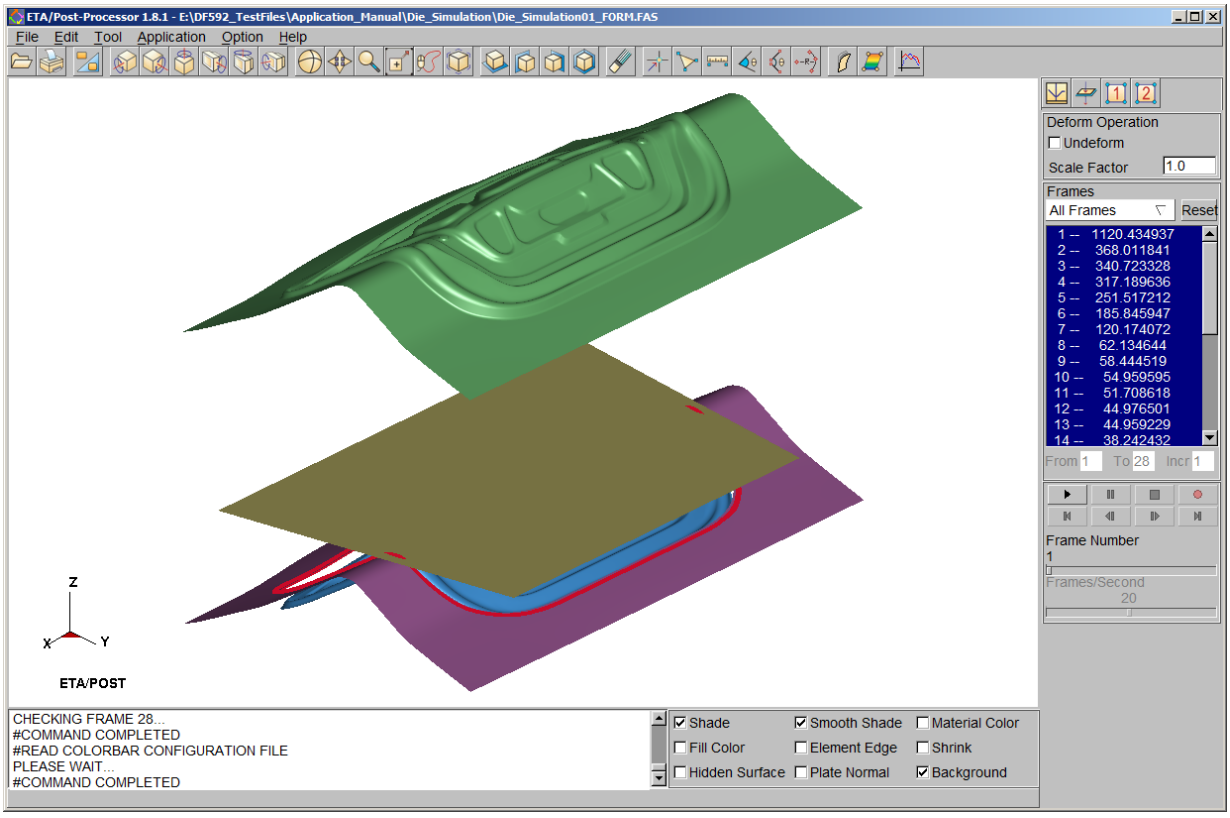


1. From the **File** menu of **eta/POST**, select **Open**. The Open File dialogue will be displayed.



2. Select **Die Simulation01_FORM.FAS** from the **Files of Type** list. This option will allow you to read in the **fas** file.

Navigate to the directory where the result files were saved, be sure you have the correct file of type selected, pick the **fas** file, and click **Open**, as shown in the figure below.



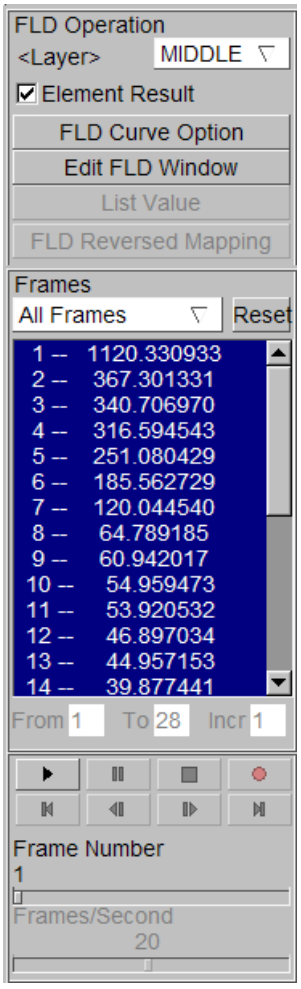
3. The **fas** file is now completely read in. You are ready to process the results using the result manipulation menu bar as shown below.

II. Animating FLD

eta/POST can animate deformation, FLD, thickness, and various strain/stress distribution of the blank. To do this, refer to the following examples.

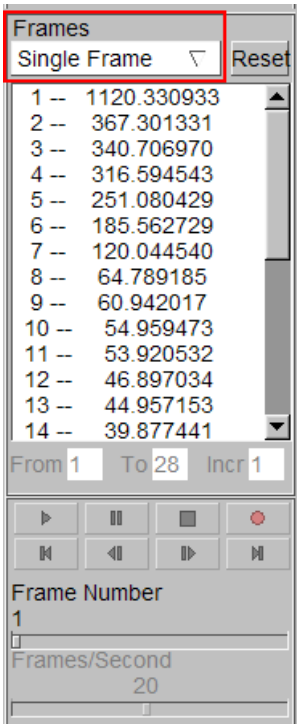
FLD

1. Pick **FLD** from the result manipulation menu bar.
2. Select **Middle** from the **Current Component** list.
3. Set FLD parameters (n, t, r, etc.) from **FLD Curve Option**.
4. Select **Edit FLD Window** to define location of FLD plot on the display window.
5. Click the **Play** button and the animation will begin.
6. Click the **Stop** button.



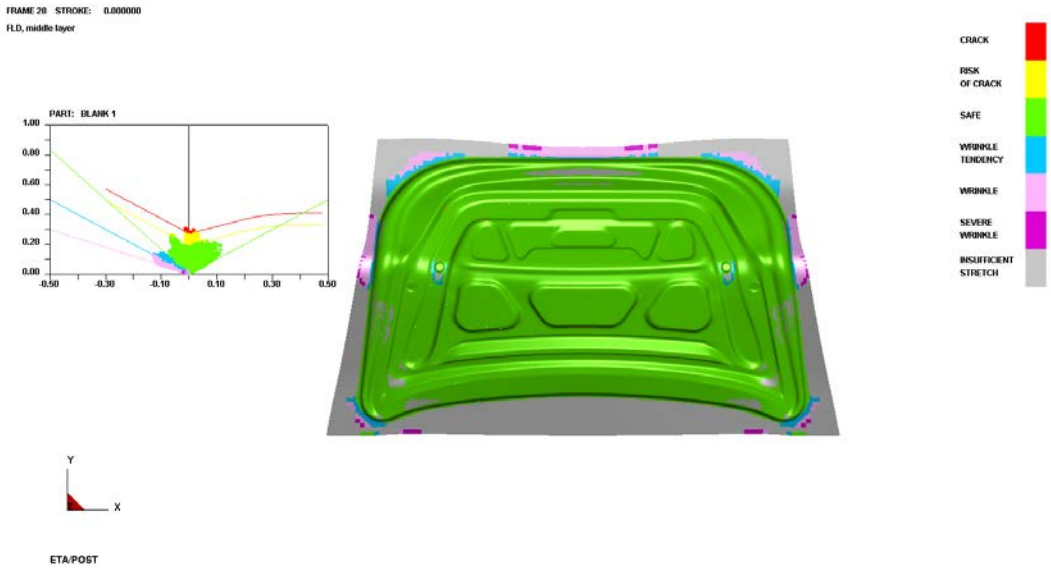
III. Plotting Single Frames

Sometimes it is more convenient to view single frames rather than the entire animation. To do this, select **Single Frame** from the **Frame** combo box. Then, select with your mouse, the frame you would like to view from the frame list. Users can also drag the slider of frame number to select the frame accordingly.



IV. Check the forming Result of FLD

In this step, we check the forming result based on the FLD plotting. Turn off all parts except **BLANK** part, and then plot the last frame, as shown below.

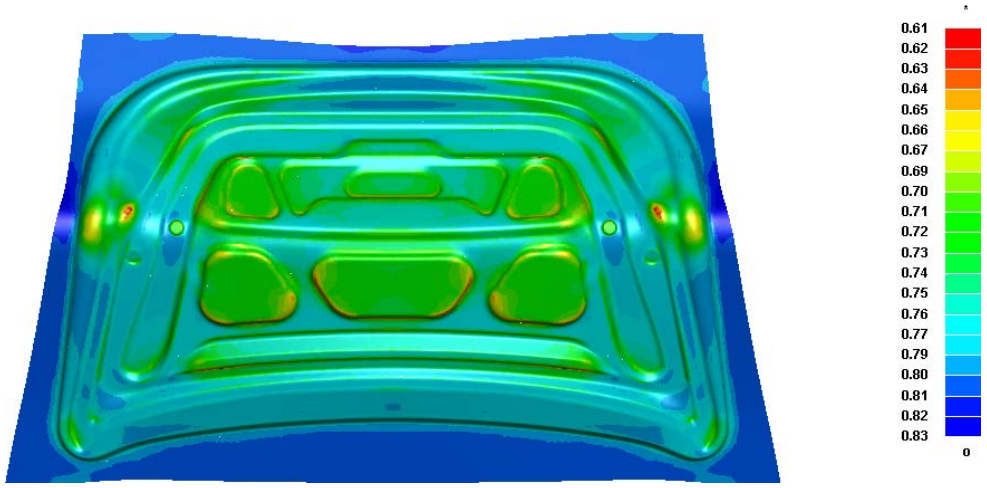


V. Check the forming Result of THICKNESS

In this step, we check the forming result based on the THICKNESS and THINNING plotting.

1. Pick **THICKNESS** from the result manipulation menu bar.

2. Select **THICKNESS** from the Current Component list.
3. Then select last frame to plot the thickness result.



4. Select **THINNING** from the Current Component list to plot thinning result.